

**B.Tech Thesis on**  
**SHELL AND TUBE HEAT EXCHANGER DESIGN USING CFD TOOLS**

*For partial fulfilment of the requirements for the degree of*

**Bachelor of Technology**  
**in**  
**Chemical Engineering**

**Submitted by**  
**Anil Kumar Samal Roll**  
**No-109CH0458**

*Under the guidance of:*

**Prof Basudeb Munshi**



Department of Chemical Engineering,  
NIT Rourkela, 769008



### **CERTIFICATE**

This is to certify that the thesis entitled “**Shell and tube heat exchanger design using CFD**” submitted by Anil kumar samal (109ch0458) in the partial fulfilment of the requirement for the award of Degree of Bachelor technology in chemical Engineering at National institute of Technology ,Rourkela is an authentic work carried out by his under my supervision and guidance.

To best of knowledge, the matter embodied in this thesis has not been submitted to any other university or institute for the award of any degree.

Date:

Place:

**Prof Basudeb Munshi**

Department of Chemical Engineering

N.I.T Rourkela , 769008

## **ACKNOWLEDGEMENT**

i would like to convey my sincere gratitude to my project supervisor Prof Basudeb Munshi for his invaluable suggestions, constructive criticism ,motivation and guidance for carrying out related experiments and for preparing the associated reports and presentations. His encouragement towards the current topic helped me a lot in this project work .

i owe my thankfulness to prof R.K.Singh , Head, Department of chemical engineering providing necessary facilities in the department .

I thank my family and friends for being constant support throughout my life .

**Date:**

**Place:**

**Anil kumar samal**

**109ch0458**

## **ABSTRACT**

In present day shell and tube heat exchanger is the most common type heat exchanger widely use in oil refinery and other large chemical process, because it suits high pressure application. The process in solving simulation consists of modeling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 13.0. The objective of the project is design of shell and tube heat exchanger with helical baffle and study the flow and temperature field inside the shell using ANSYS software tools. The heat exchanger contains 7 tubes and 600 mm length shell diameter 90 mm. The helix angle of helical baffle will be varied from  $0^0$  to  $20^0$ . In simulation will show how the pressure vary in shell due to different helix angle and flow rate. The flow pattern in the shell side of the heat exchanger with continuous helical baffles was forced to be rotational and helical due to the geometry of the continuous helical baffles, which results in a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger.

CONTENTS	Page no.
Cover page.....	i
Certificate .....	ii
Acknowledgement .....	iii
Abstract .....	iv
Content .....	v
List of Figure .....	vii
List of table .....	ix
Nomenclature .....	x
Chapter 1.....	1
1    Introduction .....	2
1.1    Objective .....	3
Chapter 2 .....	4
2.1 Literature Review .....	5
2.2 Purpose of Use of Helical Baffle .....	5
2.3 Computational Fluid Dynamics.....	6
2.4 Application Of CFD.....	7
2.5 ANSYS.....	7
Chapter 3 .....	8
3    Computational Model For Heat Exchanger .....	8

3.1 Problem Statement.....	9
3.2 Computational Model .....	9
3.3 Navier’s Stokes Equation .....	9
3.4 Geometry Of Mesh.....	10
3.5 Grid Generation .....	11
3.6 Meshing .....	12
3.7 Problem Set up .....	12
3.8 Solution Initialization .....	13
Chapter 4 .....	14
4 Results .....	15
4.1 Convergence of Simulation .....	16
4.2 Variation Of Temperature .....	18
4.3 Variation Of Velocity .....	20
4.4 Variation Of Pressure .....	21
4.5 Heat Transfer Rate.....	26
Chapter 5 .....	28
Conclusions .....	29
Chapter 6 .....	30
Reference .....	31

## **List Of Figures**

List of figures	Page no.
Fig 2.1 Fluid flow simulation for a shell and tube heat exchanger.	6
Fig 3.1 Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination.	10
Fig 3.2 complete model of shell and tube heat exchanger	12
Fig 3.3 Meshing diagram of shell and tube heat exchanger	13
Fig 4.11 For Conversion $0^0$ Baffle inclination after 170 <sup>th</sup> iteration	16
Fig 4.12 Converge simulation of $10^0$ baffle inclination at 133th iteration.	17
Fig 4.13 Convergence of $20^0$ baffle inclination at 138 <sup>th</sup> iteration	17
Fig 4.21 Temperature Distribution across the tube and shell .	18
Fig 4.22 Temperature Distribution for $10^0$ baffle inclination	18
Fig 4.23 Temperature Distribution of $20^0$ baffle inclination	19
Fig 4.24 Temperature Distribution across Tube outlet in $0^0$ baffle inclination	19
Fig 4.31 Velocity profile across the shell at $0^0$ baffle inclination.	20
Fig 4.32 Velocity profile across the shell at $10^0$ baffle inclination.	21

fig 4.41 Pressure Distribution across the shell at 0 <sup>0</sup> baffle inclination.	21
Fig 4.42 Pressure Distribution across the shell at 10 <sup>0</sup> baffle inclination	22
Fig 4.43 Pressure Distribution across the shell at 20 <sup>0</sup> baffle inclination	22
Fig 4.44 Plot of Baffle inclination angle vs Outlet Temperature of shell and tube side	23
Fig 4.45 Plot of Baffle angle vs Pressure Drop	24
Fig 4.46 Plot of Velocity profile inside shell	25
Fig 4.47 Heat Transfer Rate Along Tube side	26



## List of Tables

List of Tables	Page no
Table no 3.1 Geometric dimensions of shell and tube heat exchanger	11
Table no 4.1 for the Outlet Temperature of the Shell side And Tube Side	23
Table no 4.2 for the Pressure Drop inside Shell	24
Table no 4.3 for Velocity inside Shell	25
Table no 4.4 for Heat Transfer Rate Across Tube side	26
Table no 4.5 for the Overall Calculated value in Shell and Tube heat exchanger in this simulation.	27

## Nomenclature

L	Heat exchanger length
Di	Shell inner diameter,
do	Tube outer diameter
Nt	Number of tubes,
Nb	Number of baffles.
B	Central baffle spacing,
$\Theta$	Baffle inclination angle
Bc	Baffle cut

# **Chapter 1**

## **Introduction**

# **1.INTRODUCTION**

Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their application. For example, heat exchangers being used to condense are known as condensers, similarly heat exchanger for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transfer using least area of heat transfer and pressure drop. A more better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements.

Heat exchangers are of two types:-

- Where both media between which heat is exchanged are in direct contact with each other is Direct contact heat exchanger,
- Where both media are separated by a wall through which heat is transferred so that they never mix, Indirect contact heat exchanger.

A typical heat exchanger, usually for higher pressure applications up to 552 bars, is the shell and tube heat exchanger. Shell and tube type heat exchanger, indirect contact type heat exchanger. It consists of a series of tubes, through which one of the fluids runs. The shell is the container for the shell fluid. Generally, it is cylindrical in shape with a circular cross section, although shells of different shape are used in specific applications. For this particular study shell is considered, which is generally a one pass shell. A shell is the most commonly used due to its low cost and simplicity, and has the highest log-mean temperature-difference (LMTD) correction factor. Although the tubes may have single or multiple passes, there is one pass on the shell side, while the other fluid flows within the shell over the tubes to be heated or cooled. The tube side and shell side fluids are separated by a tube sheet.

Baffles are used to support the tubes for structural rigidity, preventing tube vibration and sagging and to divert the flow across the bundle to obtain a higher heat transfer coefficient. Baffle spacing ( $B$ ) is the centre line distance between two adjacent baffles, Baffle is provided with a cut ( $B_c$ ) which is expressed as the percentage of the segment height to shell inside diameter. Baffle cut can vary between 15% and 45% of the shell inside diameter. In the present study 36% baffle cut ( $B_c$ ) is considered. In general, conventional shell and tube heat exchangers result in high shell-side pressure drop and formation of recirculation zones near the baffles. Most of the researches now a day are carried on helical baffles, which give better performance than single segmental baffles but they involve high manufacturing cost, installation cost and maintenance cost. The effectiveness and cost are two important parameters in heat exchanger design. So, In order to improve the thermal performance at a reasonable cost of the Shell and tube heat exchanger, baffles in the present study are provided with some inclination in order to maintain a reasonable pressure drop across the exchanger.

The complexity with experimental techniques involves quantitative description of flow phenomena using measurements dealing with one quantity at a time for a limited range of problem and operating conditions. Computational Fluid Dynamics is now an established industrial design tool, offering obvious advantages. In this study, a full 360° CFD model of shell and tube heat exchanger is considered. By modelling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained.

## **1.1OBJECTIVE:**

The main objective of this project is designing and simulation of shell and tube heat exchanger with helical baffle using Ansys tools.

# **Chapter 2**

## **Literature Review**

## **2.LITERATURE REVIEW**

### **2.1 Introduction**

The purpose of this chapter is to provide a literature review of past research effort such as journals or articles related to shell and tube heat exchanger and computational fluid dynamics (CFD) analysis whether on two dimension and three dimension modelling. Moreover, review of other relevant research studies are made to provide more information in order to understand more on this research.

### **2.2 Purpose of Use of Helical Baffle:**

A new type of baffle, called the helical baffle, provides further improvement. This type of baffle was first developed by Lutchka and Nemcansky. They investigated the flow field patterns produced by such helical baffle geometry with different helix angles. They found that these flow patterns were very close to the plug flow condition, which was expected to reduce shell-side pressure drop and to improve heat transfer performance. Stehlik et al. compared heat transfer and pressure drop correction factors for a heat exchanger with an optimized segmental baffle based on the Bell–Delaware method, with those for a heat exchanger with helical baffles. Kral et al. discussed the performance of heat exchangers with helical baffles based on test results of various baffles geometries. One of the most important Geometric factors of the STHXHB is the helix angle. Recently a comprehensive comparison between the test data of shell-side heat transfer coefficient versus shell-side pressure drop was provided for five helical baffles and one segmental baffle measured for oil-water heat exchanger. It is found that based on the heat transfer per unit shell-side fluid pumping power or unit shell-side fluid pressured drop, the case of 40<sup>0</sup> helix angle behaves the best. The flow pattern in the shell side of the heat exchanger with continuous helical baffles was forced to be rotational and helical due to the geometry of the continuous helical baffles, which results in a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger. Properly designed continuous helical baffles can reduce fouling in the shell side and prevent the flow-induced vibration as well. The performance of the proposed STHXs was studied experimentally in this work. The use of continuous helical baffles results in nearly 10% increase in heat transfer coefficient compared with that of conventional segmental baffles for the same shell-side pressure drop. Based on the experimental data, the non dimensional correlations for heat transfer coefficient and pressure

drop were developed for the proposed continuous helical baffle heat exchangers with different shell configurations, which might be useful for industrial applications and further study of continuous helical baffle heat exchangers.

## **2.3 Computational Fluid Dynamics (CFD):**

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modelling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design.

Computational fluid dynamics (CFD) is useful in a wide variety of applications and use in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyse problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are on going research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger (Figure 1). Furthermore, motor vehicle manufactures now routinely predict drag forces, underbonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

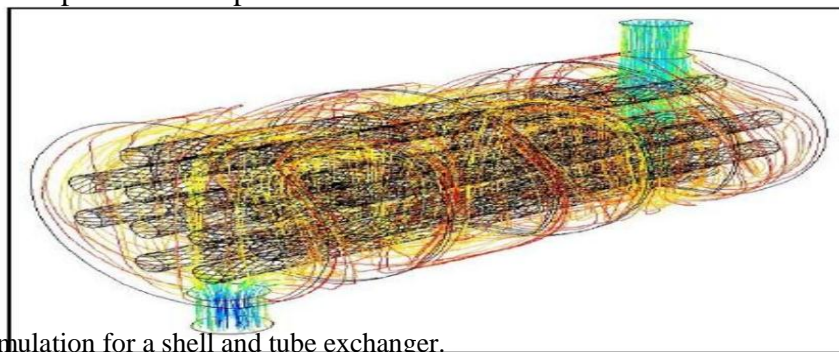


fig 2.1 Fluid flow simulation for a shell and tube exchanger.



## **2.4 APPLICATION OF CFD:**

CFD not just spans on chemical industry, but a wide range of industrial and nonindustrial application areas which is in below :

- Aerodynamics of aircraft and vehicle.
- Combustion in IC engines and gas turbine in power plant.
- Loads on offshore structure in marine engineering.
- Blood flows through arteries and vein in biomedical engineering.
- Weather prediction in meteorology.
- Flow inside rotating passages and diffusers in turbo-machinery.
- External and internal environment of buildings like wind loading and heating or
- Ventilation system.
- Mixing and separation or polymer moldings in chemical process engineering.
- Distribution of pollutants and effluent in environmental engineering.

## **2.5 ANSYS:**

Ansys is the finite element analysis code widely use in computer aided engineering(CAE) field. ANSYS software help us to construct computer models of structure, machine, components or system, apply operating loads and other design criteria, study physical response such as stress level temperature distribution, pressure etc.

In Ansys following Basic step is followed:

- During pre processing the geometry of the problem is defined. Volume occupied by fluid is divided into discrete cells(the mesh). The mesh may be uniform or non uniform. The physical modelling is defined. Boundry condition is defined. This involves specifying the fluid behaviour of the problem. For transient problem boundry condition are also defined.
- The simulation is started and the equation are solved iteratively as steady state or transient.
- Finally a post procedure is used for the analysis and visualisation of the resulting problem.

# **Chapter 3**

## **COMPUTATIONAL MODEL FOR HEAT EXCHANGER**

### **3. COMPUTATIONAL MODEL FOR HEAT EXCHANGER**

#### **3.1 Problem Description:**

Design of shell and tube heat exchanger with helical baffle using CFD. To study the temperature and pressure inside the tube with different mass flow rate .

#### **3.2 Computational Model:**

The computational model of an experimental tested Shell and Tube Heat Exchanger(STHX) with 10 helix angle is shown in fig. 2, and the geometry parameters are listed in Table 1.As can be seen from Fig 2 ,the simulated STHX has six cycles of baffles in the shell side direction with total number of tube 7 .The whole computation domain is bounded by the inner side of the shell and everything in the shell contained in the domain. The inlet and outlet of the domain are connected with the corresponding tubes.

To simplify numerical simulation, some basic characteristics of the process following assumption are made :

1. The shell side fluid is constant thermal properties
2. The fluid flow and heat transfer processes are turbulent and in steady state
3. The leak flows between tube and baffle and that between baffles and shell are neglected
4. The natural convection induced by the fluid density variation is neglected
5. The tube wall temperature kept constant in the whole shell side
6. The heat exchanger is well insulated hence the heat loss to the environment is totally neglected .

#### **3.3 Navier-Stokes Equation:**

It is named after Claude-Louis Navier and Gabriel Stokes , He described the motion of fluid substances. Its also a fundamental equation being used by ANSYS and even in the present project work. These equation arise from applying second law of newton to fluid motion, together with the assumption that the fluid stress is sum of a diffusing viscous term ,plus a pressure term. The derivation of the Navier Stokes equation begins with an application of second law of newton i.e conservation of momentum. In an inertial frame of reference, the

general form of the equations of fluid motion is :-

$$\partial_x u + \partial_y v = 0, \quad (1)$$

$$\begin{aligned} \partial_t u + u \partial_x u + v \partial_y u = -\partial_x p + \frac{1}{\text{Re}} [\partial_x (\mu \partial_x u) + \partial_y (\mu \partial_y u) \\ + \partial_x \mu \partial_x u + \partial_y \mu \partial_x v], \end{aligned} \quad (2)$$

$$\begin{aligned} \partial_t v + u \partial_x v + v \partial_y v = -\partial_y p + \frac{1}{\text{Re}} [\partial_x (\mu \partial_x v) + \partial_y (\mu \partial_y v) \\ + \partial_y \mu \partial_y v + \partial_x \mu \partial_y u], \end{aligned} \quad (3)$$

$$\partial_t T + u \partial_x T + v \partial_y T = -\frac{1}{\text{Re Pr}} [\partial_x (\kappa \partial_x T) + \partial_y (\kappa \partial_y T)], \quad (4)$$

This Navier Stokes Equation solve in every mesh shell and the simulation shows the result.

### 3.4 Geometry and Mesh:

The model is designed according to TEMA (Tubular Exchanger Manufacturers Association) Standards Gaddis (2007).

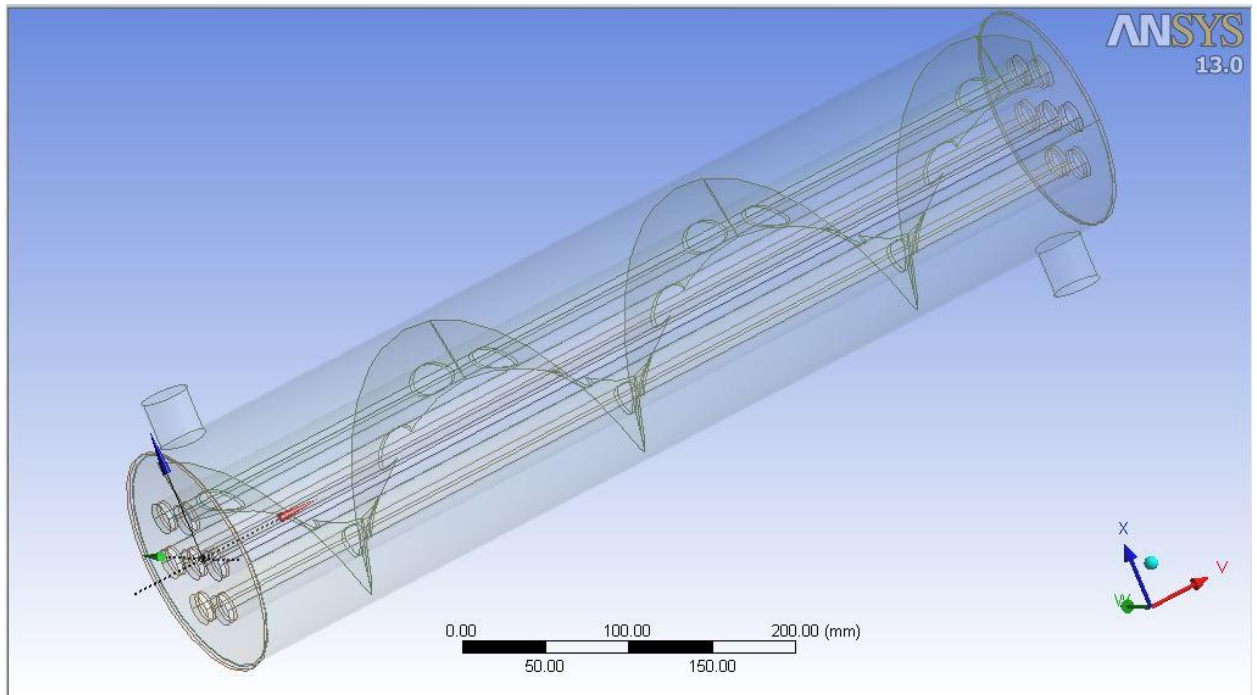


Fig 3.1 Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination.

**Table 3.1 Geometric dimensions of shell and tube heat exchanger**

Heat exchanger length, $L$	600mm
Shell inner diameter, $D_i$	90mm
Tube outer diameter, $d_o$	20mm
Tube bundle geometry and pitch Triangular	30mm
Number of tubes, $N_t$	7
Number of baffles, $N_b$	6
Central baffle spacing, $B$	86mm
Baffle inclination angle, $\theta$	0 to 40°

### 3.5. Grid Generation

The three-dimensional model is then discretized in ICEM CFD. In order to capture both the thermal and velocity boundary layers the entire model is discretized using hexahedral mesh elements which are accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The entire geometry is divided into three fluid domains Fluid\_Inlet, Fluid\_Shell and Fluid\_Outlet and six solid domains namely Solid\_Baffle1 to Solid\_Baffle6 for six baffles respectively.

The heat exchanger is discretized into solid and fluid domains in order to have better control over the number of nodes. The fluid mesh is made finer than solid mesh for simulating conjugate heat transfer phenomenon. The three fluid domains are as shown in Fig.

2. The first cell height in the fluid domain from the tube surface is maintained at 100 microns to capture the velocity and thermal boundary layers. The discretised model is checked for quality and is found to have a minimum angle of 18° and min determinant of 4.12. Once the meshes are checked for free of errors and minimum required quality it is exported to ANSYS CFX pre-processor.

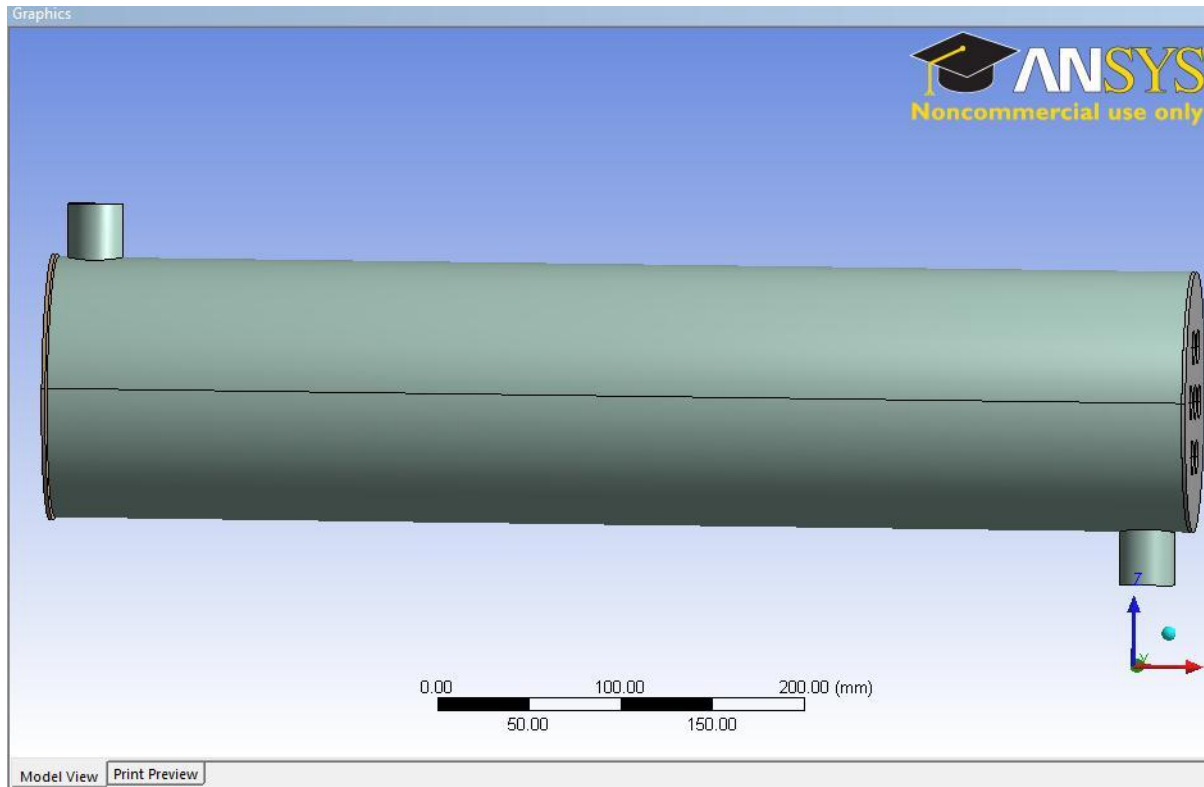


Fig 3.2 complete model of shell and tube heat exchanger

### 3.6 Meshing :

Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 5.65 Million cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

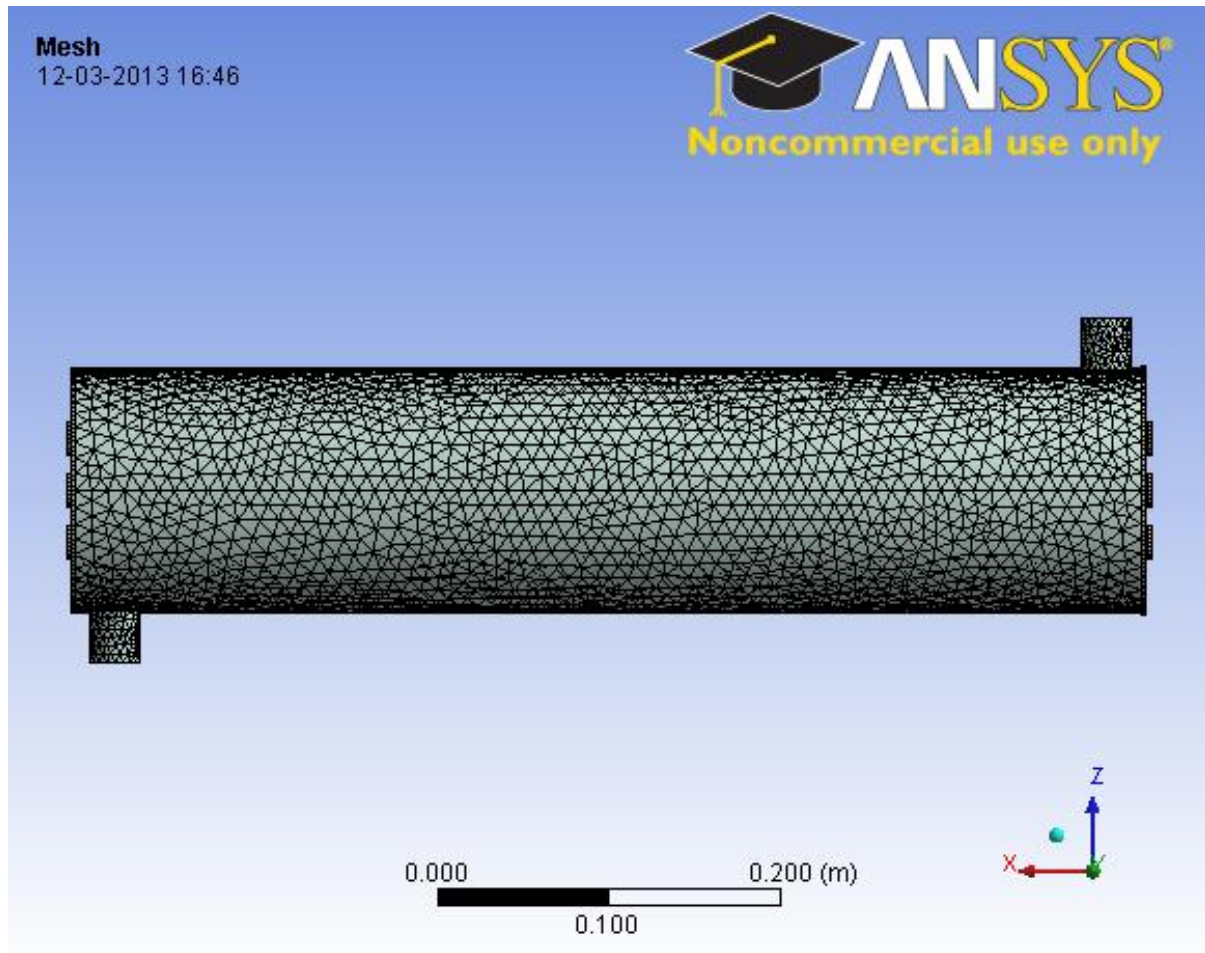


Fig 3.3 Meshing diagram of shell and tube heat exchanger

### 3.7 Problem Setup

Simulation was carried out in ANSYS® FLUENT® v13. In the Fluent solver Pressure Based type was selected , absolute velocity formation and steady time was selected for the simulation . In the model option energy calculation was on and the viscous was set as standard k-e, standard wall function(k-epsilon 2 eqn).

In cell zone fluid water-liquid was selected. Water-liquid and copper, aluminum was selected as materials for simulation. Boundary condition was selected for inlet,outlet. In inlet and outlet 1kg/s velocity and temperature was set at 353k. Across each tube 0.05kg/s velocity and 300k temperature was set. Mass flow was selected in each inlet. In reference Value Area set as  $1\text{m}^2$  ,Density  $998\text{ kg/m}^3$  ,enthalpy  $229485\text{ j/kg}$  , length  $1\text{m}$  , temperature  $353\text{k}$ , Velocity  $1.44085\text{ m/s}$  , Ration of specific heat 1.4 was considered .

### **3.8 Solution Initialization:**

Pressure Velocity coupling selected as SIMPLEC. Skewness correction was set at zero. In Spatial Discretization zone Gradient was set as Least square cell based , Pressure was standard , Momentum was First order Upwind , Turbulent Kinetic energy was set as First order Upwind , Energy was also set as First order Upwind. In Solution control, Pressure was 0.7, Density 1 , Body force 1, Momentum 0.2 , turbulent kinetic and turbulent dissipation rate was set at 1, energy and turbulent Viscosity was 1. Solution initialization was standard method and solution was initialize from inlet with 300k temperature.



# **Chapter 4**

## **Results**

## 4 Results

Under the Above boundary condition and solution initialize condition simulation was set for 1000 iteration.

### **4.1 Convergence Of Simulation :**

The convergence of Simulation is required to get the parameters of the shell and tube heat exchanger in outlet. It also gives accurate value of parameters for the requirement of heat transfer rate. Continuity, X-velocity, Y-velocity, Z-velocity, energy, k , epsilon are the part of scaled residual which have to converge in a specific region. For the continuity,X-velocity ,Y-velocity, Z-velocity , k, epsilon should be less than  $10^{-4}$  and the energy should be less than  $10^{-7}$ . If these all values in same manner then solution will be converged.

$0^0$  Baffle inclination

For Zero degree baffle inclination solution was converged at 170<sup>th</sup> iteration. The following figure shows the residual plot for the above iterations:

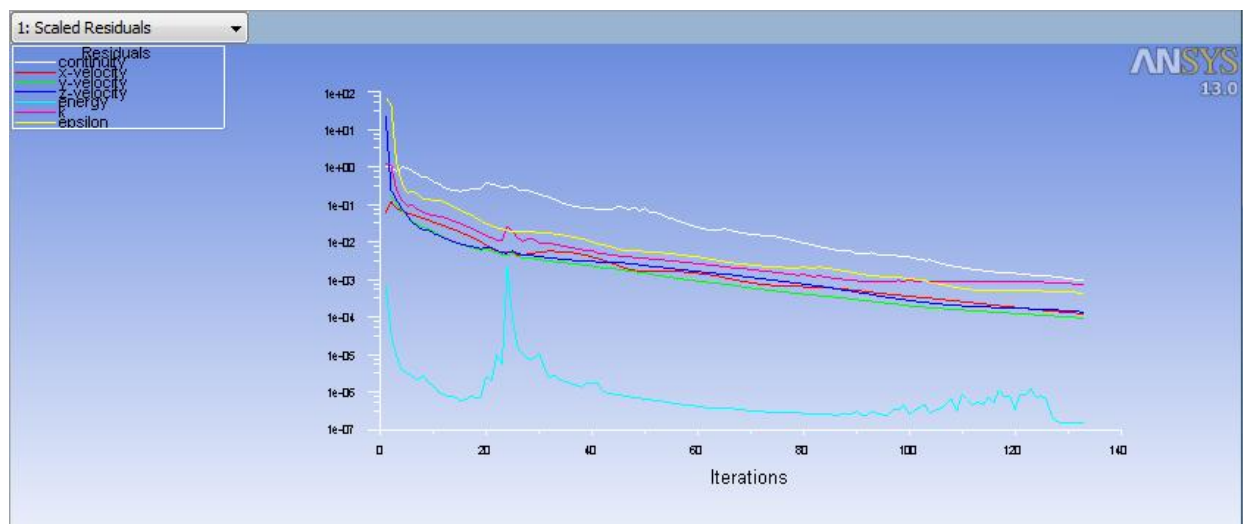


Figure 4.11- For Conversion  $0^0$  Baffle inclination after 170<sup>th</sup> iteration

10<sup>0</sup> Baffle inclination:

Simulation of 10<sup>0</sup> Baffle inclination is converged at 133th iteration. The following figure shows the residual plot:

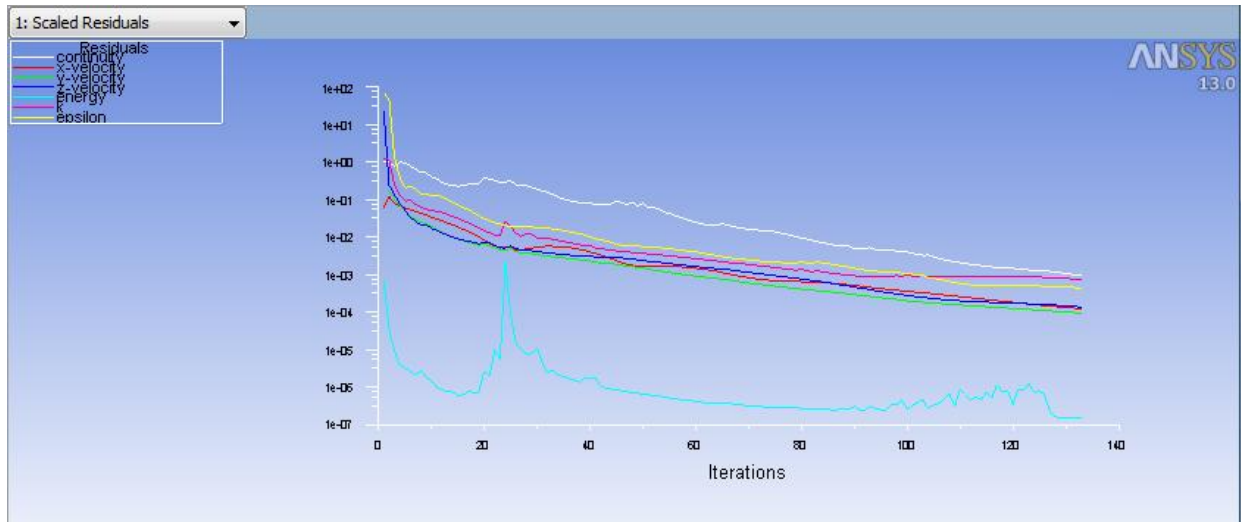


Figure 4.12 Converge simulation of 10<sup>0</sup> baffle inclination at 133th iteration.

20<sup>0</sup> Baffle inclination:

Simulation of 20<sup>0</sup> baffle inclination is converged at 138<sup>th</sup> iteration. The following figure shows the residual plot:

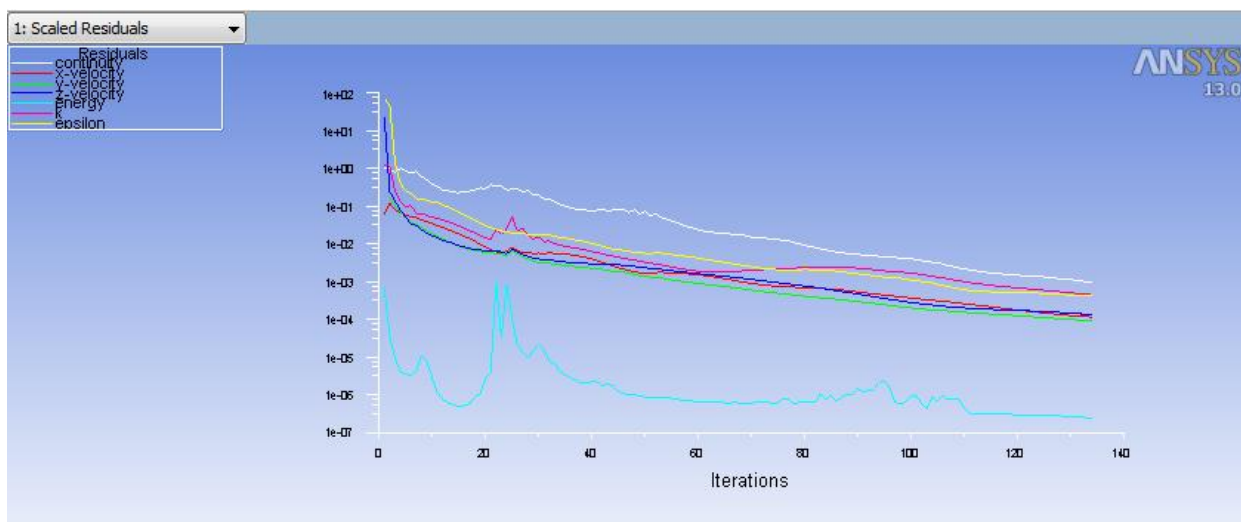


Figure 4.13 Convergence of 20<sup>0</sup> baffle inclination at 138<sup>th</sup> iteration

## 4.2 Variation of Temperature:

The temperature Contours plots across the cross section at different inclination of baffle along the length of heat exchanger will give an idea of the flow in detail. Three different plots of temperature profile are taken in comparison with the baffle inclination at  $0^\circ$ ,  $10^\circ$ ,  $20^\circ$ .

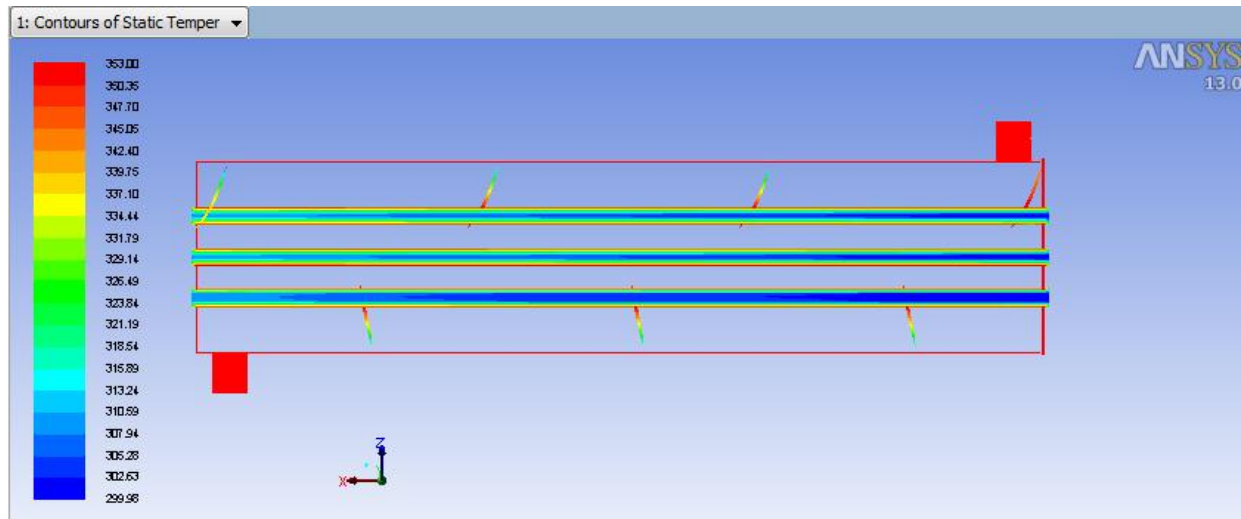


Figure 4.21 Temperature Distribution across the tube and shell .

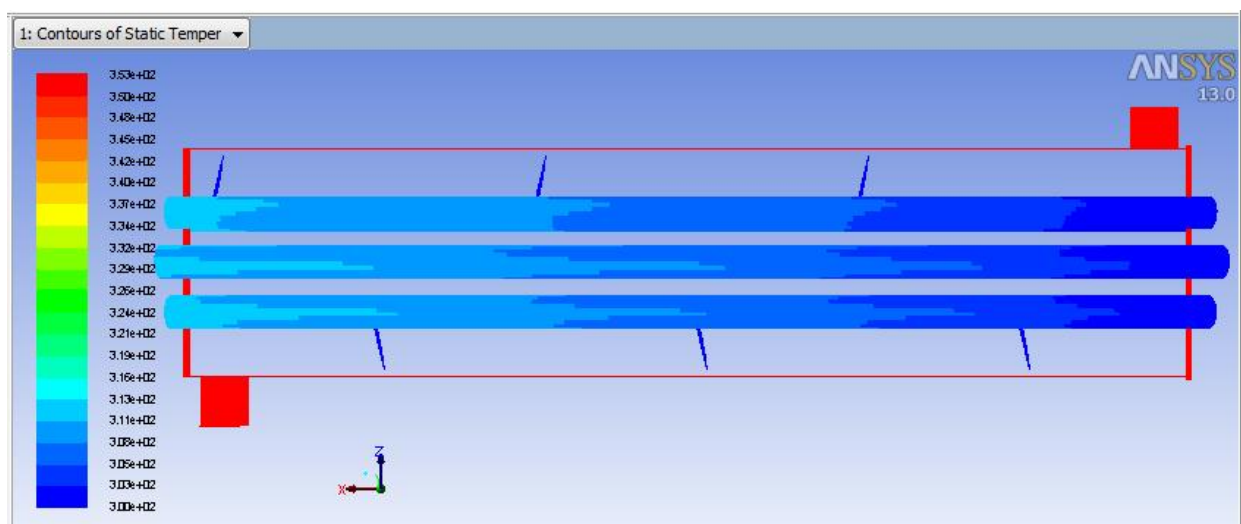


Figure 4.22 Temperature Distribution for  $10^\circ$  baffle inclination

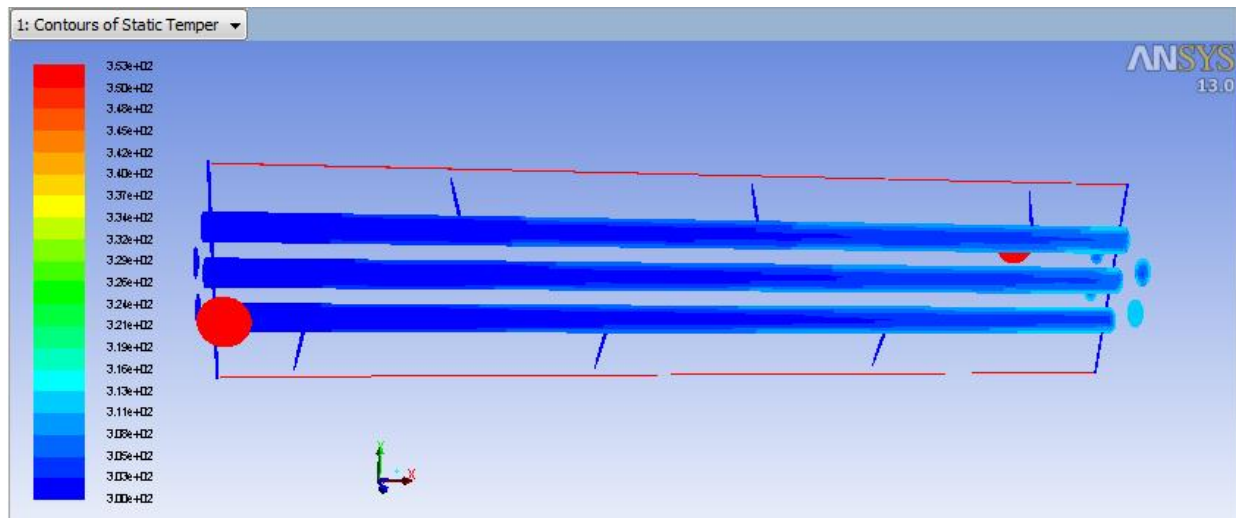


Figure 4.23 Temperature Distribution of 20<sup>0</sup> baffle inclination

Temperature of the hot water in shell and tube heat exchanger at inlet was 353k and in outlet it became 347k. In case of cold water inlet temperature was 300k and the outlet became 313k. Tube outlet Temperature Distribution was given below :

Exchanger

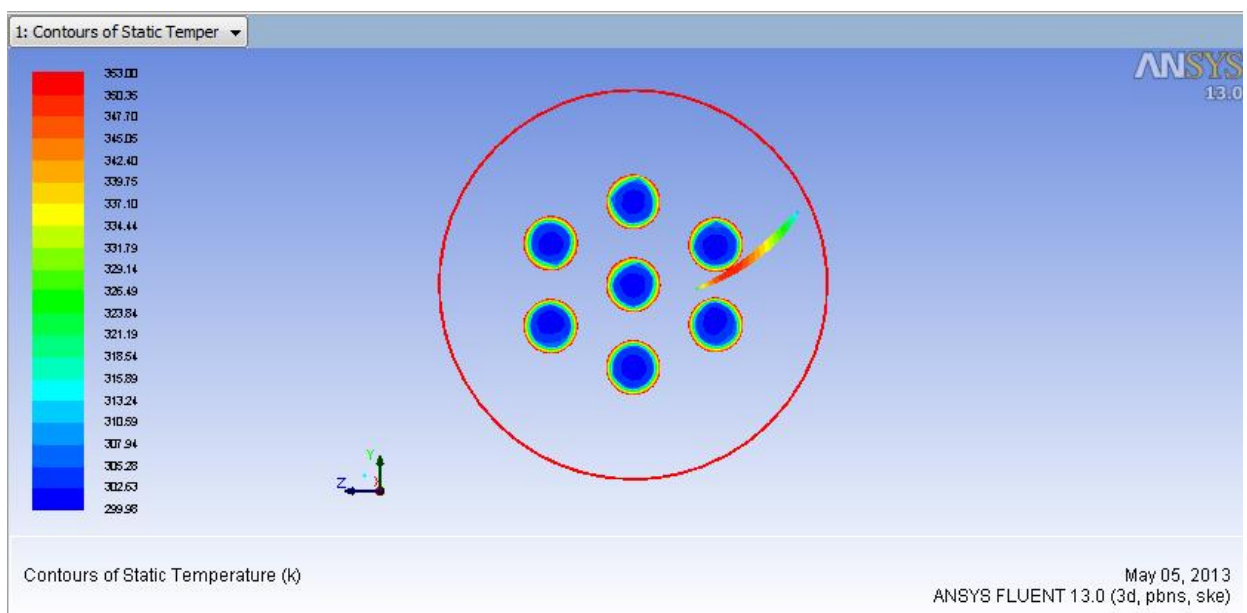


Figure 4.24 Temperature Distribution across Tube outlet in 0<sup>0</sup> baffle inclination

### 4.3 Variation Of Velocity:

Velocity profile is examined to understand the flow distribution across the cross section at different positions in heat exchanger. Below in Figure (12) (13) (14) is the velocity profile of Shell and Tube Heat exchanger at different Baffle inclination. It should be kept in mind that the heat exchanger is modeled considering the plane symmetry. The velocity profile at inlet is same for all three inclination of baffle angle i.e 1.44086 m/s. Outlet velocity vary tube to helical baffle and turbulence occur in the shell region.

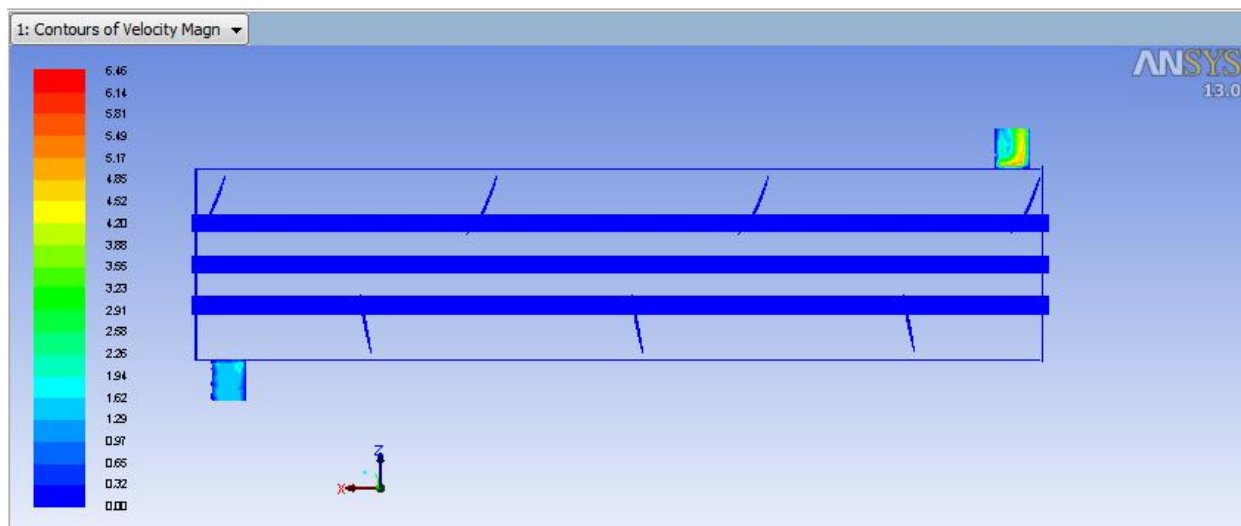


Figure 4.31 Velocity profile across the shell at  $0^\circ$  baffle inclination.

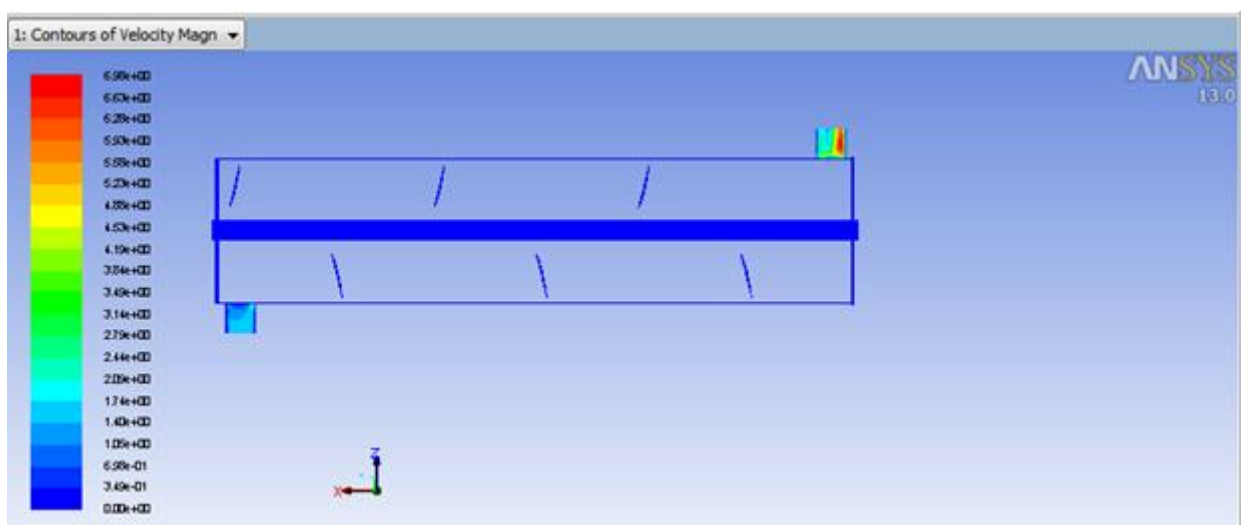


Figure 4.32 Velocity profile across the shell at  $10^0$  baffle inclination.

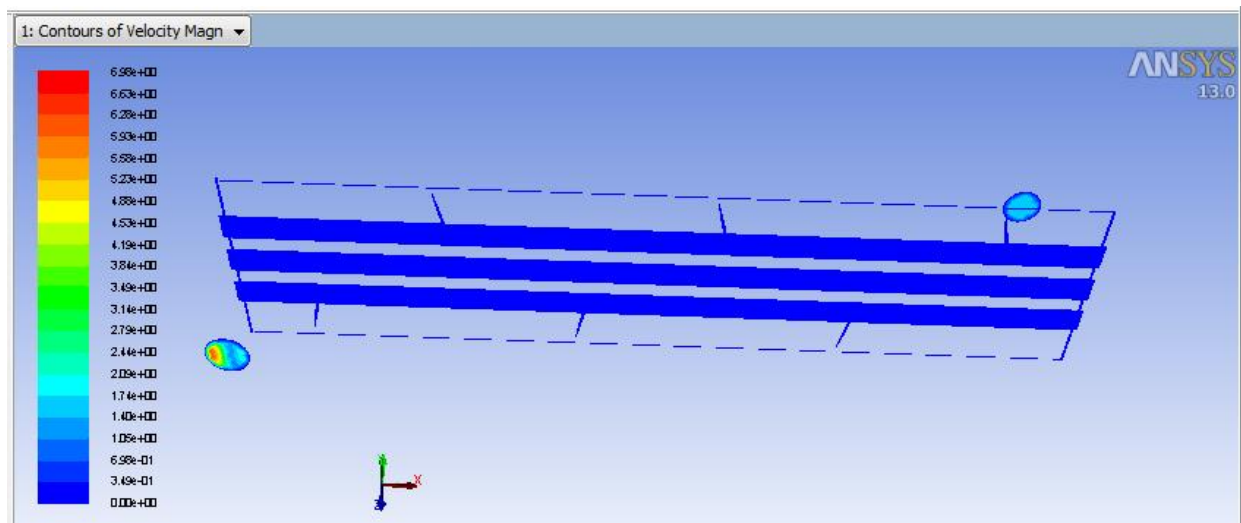


Figure 4.33 Velocity profile across the shell at  $20^0$  baffle inclination.

#### 4.4 Variation Of Pressure:

Pressure Distribution across the shell and tube heat exchanger is given below in Fig. (14) (15) (16). With the increase in Baffle inclination angle pressure drop inside the shell is decrease. Pressure vary largely from inlet to outlet. The contours of static pressure is shown in all the figure to give a detail idea.

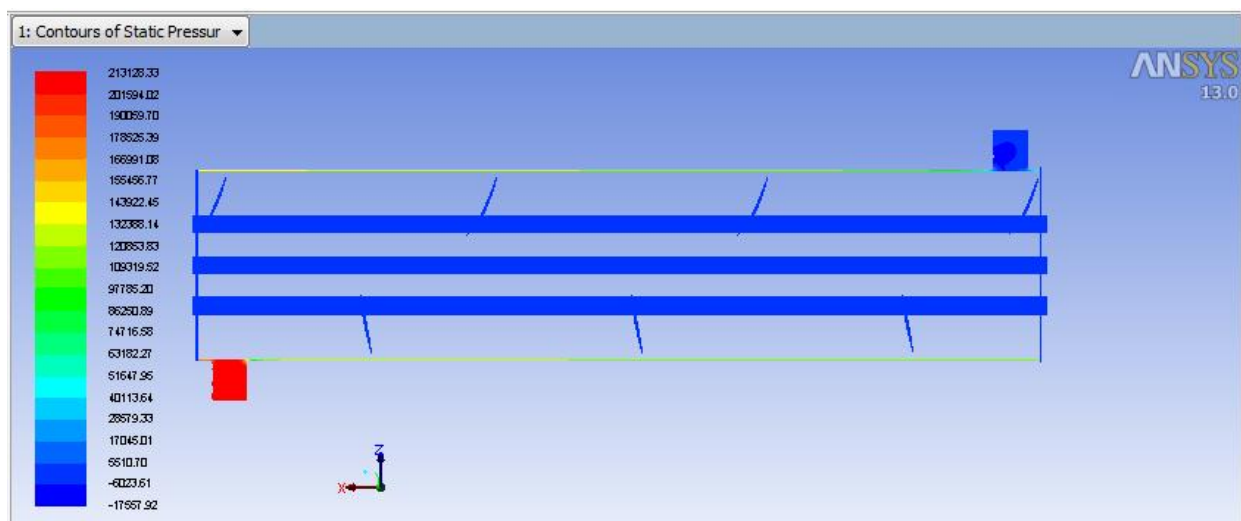


Figure 4.41 Pressure Distribution across the shell at  $0^0$  baffle inclination.

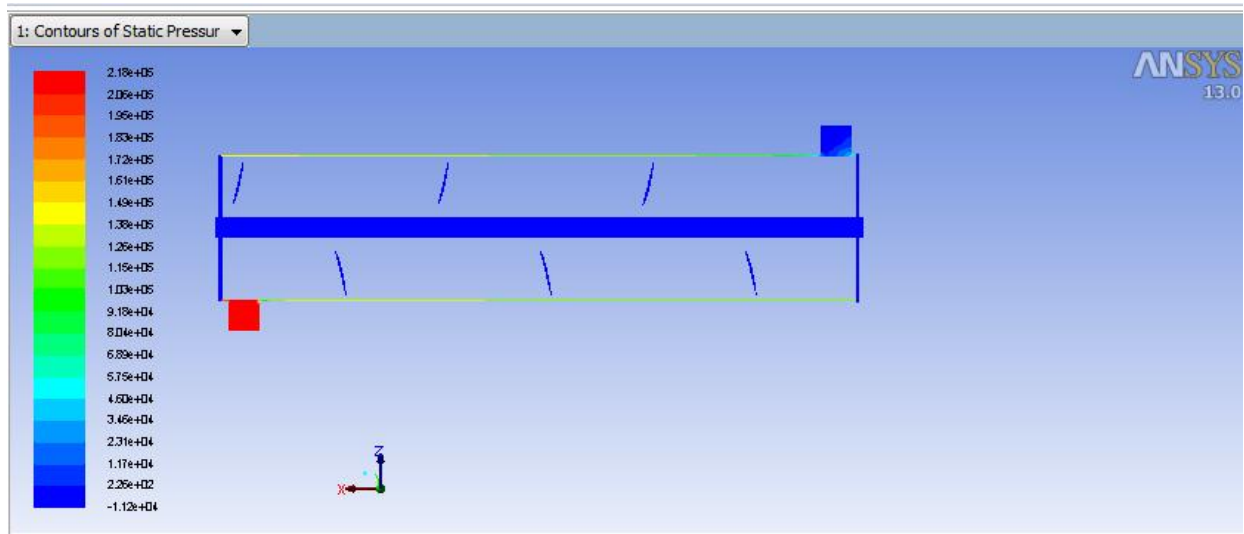


Figure 4.42 Pressure Distribution across the shell at 10<sup>0</sup> baffle inclination

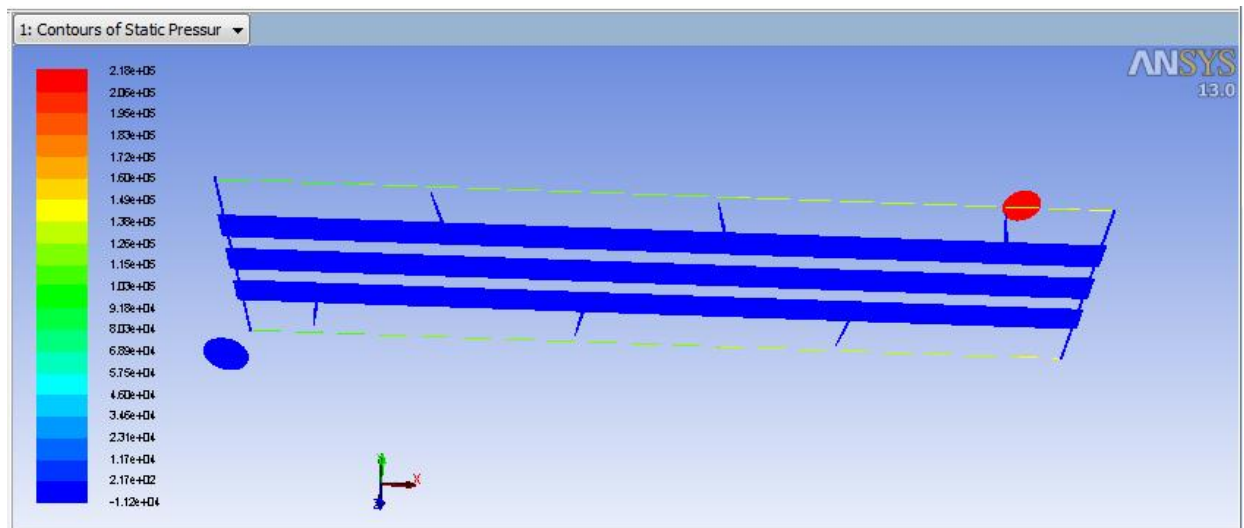


Figure 4.43 Pressure Distribution across the shell at 20<sup>0</sup> baffle inclination.



Table 4.1 for the Outlet Temperature of the Shell side And Tube Side

Baffle Inclination Angle (Degree)	Outlet Temperature Of Shell side	Outlet Temperature Of Tube side
0	346	317
10	347.5	319
20	349	320

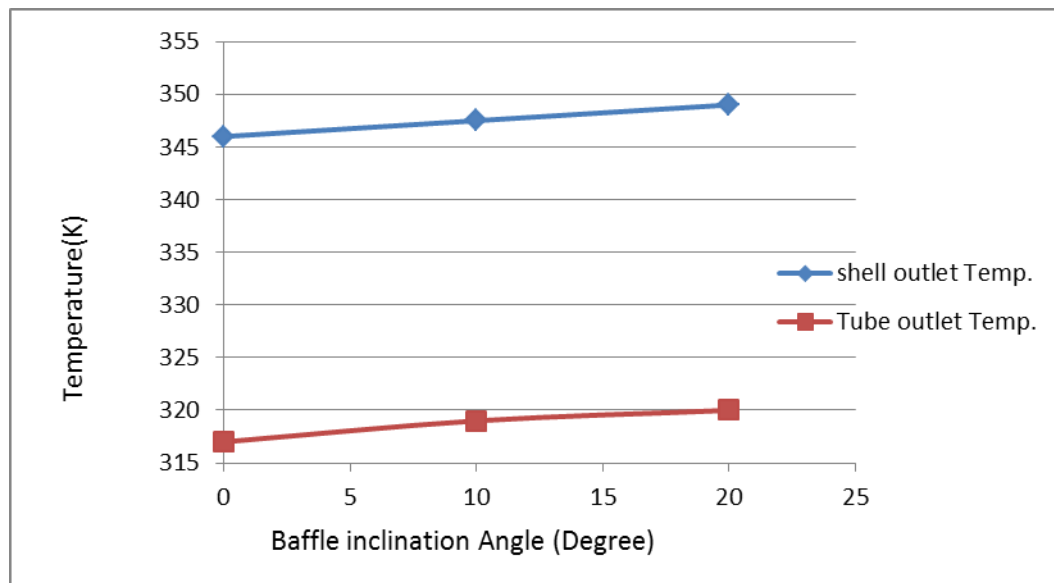


Figure 4.44 Plot of Baffle inclination angle vs Outlet Temperature of shell and tube side

It has been found that there is much effect of outlet temperature of shell side with increasing the baffle inclination angle from  $0^0$  to  $20^0$ .

Table 4.2 for the Pressure Drop inside Shell

Baffle Inclination Angle (Degree)	Pressure Drop Inside Shell (kPa)
0	230.992
10	229.015
20	228.943

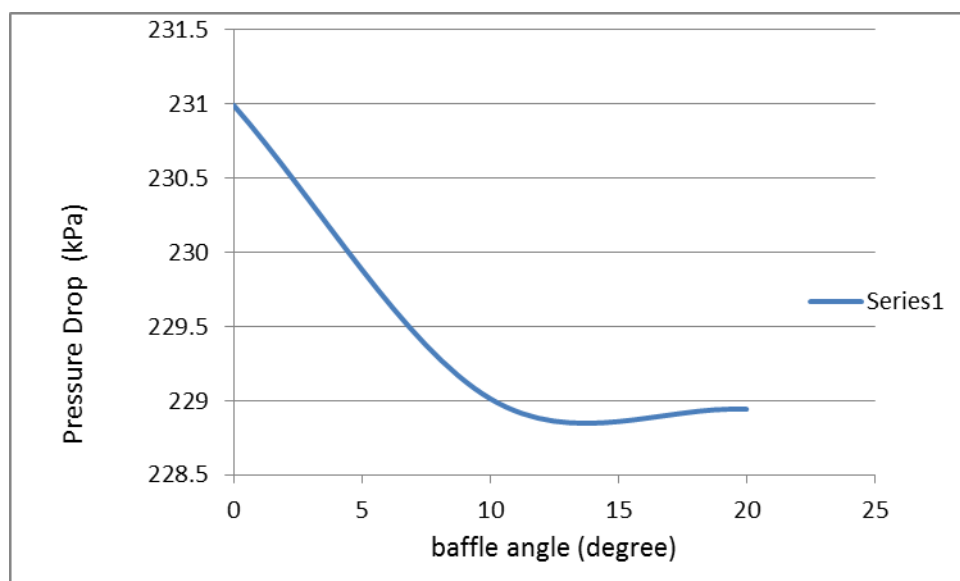


Figure 4.45 Plot of Baffle angle vs Pressure Drop

The shell-side pressure drop is decreased with increase in baffle inclination angle i. e., as the inclination angle is increased from 0° to 20°. The pressure drop is decreased by 4 %, for heat exchanger with 10° baffle inclination angle and by 16 % for heat exchanger with 20° baffle

inclination compared to  $0^\circ$  baffle inclination heat exchanger as shown in fig. 18. Hence it can be observed with increasing baffle inclination pressure drop decreases, so that it affect in heat transfer rate which is increased.

Table 4.3 for Velocity inside Shell

Baffle Inclination Angle (Degree)	Velocity inside shell (m/sec)
0	4.2
10	5.8
20	6.2

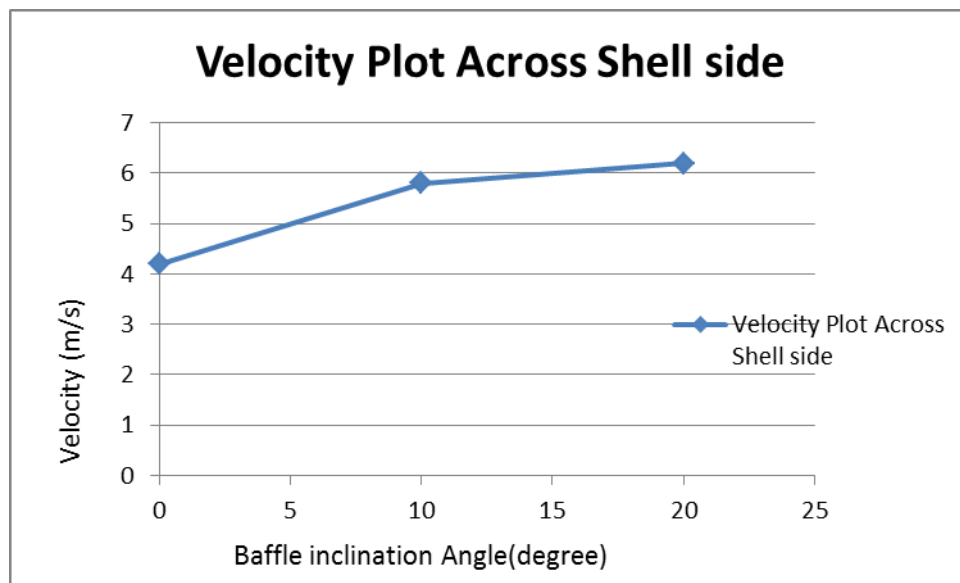


Figure 4.46 Plot of Velocity profile inside shell

The outlet velocity is increasing with increase in baffle inclination. So that more will be heat transfer rate with increasing velocity.

## 4.5 Heat Transfer Rate

$$Q = m * C_p * \Delta T$$

m=mass flow rate

$C_p$  = Specific Heat of Water

$\Delta T$  = Temperature Difference Between Tube Side

Table 4.4 for Heat Transfer Rate Across Tube side

Baffle Inclination Angle (Degree)	Heat Transfer Rate Across Tube side (w/m <sup>2</sup> )
0	3557.7
10	3972.9
20	4182

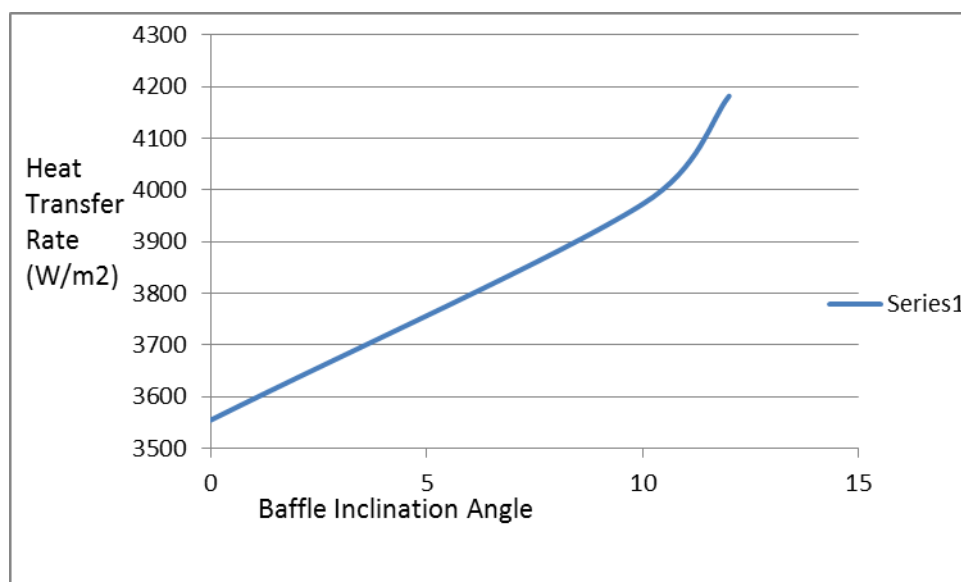


Figure 4.47 Heat Transfer Rate Along Tube side

The heat transfer rate is calculated from above formulae from which heat transfer rate is calculated across shell side. The Plot showing the with increasing baffle inclination heat transfer

rate increase. For better heat transfer rate helical baffle is used and the resulting is shown in figure 20.

Table 4.5 for the Overall Calculated value in Shell and Tube heat exchanger in this simulation.

Baffle inclination (in Degree)	Shell Outlet Temperature	Tube Outlet Temperature	Pressure Drop	Heat Transfer Rate(Q) (in W/m <sup>2</sup> )	Outlet Velocity(m/s)
0 <sup>0</sup>	346	317	230.992	3554.7	4.2
10 <sup>0</sup>	347.5	319	229.015	3972.9	5.8
20 <sup>0</sup>	349	320	228.943	4182	6.2

- The shell side of a small shell-and-tube heat exchanger is modeled with sufficient detail to resolve the flow and temperature fields.
- The pressure drop decreases with increase in baffle inclination.
- The heat transfer rate is very slow in this model so that it affect the outlet temperature of the shell and tube side.

# **Chapter 5**

## **Conclusions**

## **5 Conclusions**

The heat transfer and flow distribution is discussed in detail and proposed model is compared With increasing baffle inclination angle. The model predicts the heat transfer and pressure drop with an average error of 20%. Thus the model can be improved. The assumption worked well in this geometry and meshing expect the outlet and inlet region where rapid mixing and change in flow direction takes place. Thus improvement is expected if the helical baffle used in the model should have complete contact with the surface of the shell, it will help in more turbulence across shell side and the heat transfer rate will increase. If different flow rate is taken, it might be help to get better heat transfer and to get better temperature difference between inlet and outlet. Moreover the model has provided the reliable results by considering the standard k-e and standard wall function model, but this model over predicts the turbulence in regions with large normal strain. Thus this model can also be improved by using Nusselt number and Reynolds stress model, but with higher computational theory. Furthermore the enhance wall function are not use in this project, but they can be very useful. The heat transfer rate is poor because most of the fluid passes without the interaction with baffles. Thus the design can be modified for better heat transfer in two ways either the decreasing the shell diameter, so that it will be a proper contact with the helical baffle or by increasing the baffle so that baffles will be proper contact with the shell. It is because the heat transfer area is not utilized efficiently. Thus the design can further be improved by creating cross-flow regions in such a way that flow doesn't remain parallel to the tubes. It will allow the outer shell fluid to have contact with the inner shell fluid, thus heat transfer rate will increase.

# Chapter 6

## Reference



## **6 References**

1. Emerson, W.H., “Shell-side pressure drop and heat transfer with turbulent flow in segmentally baffled shell-tube heat exchangers”, *Int. J. Heat Mass Transfer* 6 (1963), pp. 649–66.
2. Haseler, L.E., Wadeker, V.V., Clarke, R.H., (1992), "Flow Distribution Effect in a Plate and Frame Heat Exchanger", *ICHEME Symposium Series* , No. 129, pp. 361-367.
3. Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", *Proc. 9th Proc. 9th Heat Transfer Conf.*, Israel, pp. 235-240.
4. Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , “ 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle”, a journal ,School of energy and power engineering,china.
5. Li, H., Kottke, “Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement”, source book on *Int. J. Heat Mass Transfer* 41 (1998), 10, pp. 1303–1311.
6. Thirumarimurugan, M., Kannadasan, T., Ramasamy, E., Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, *American Journal of Applied Sciences* 5 (2008), pp. 548-552.
7. Usman Ur Ehman , Göteborg, Sweden 2011, Master’s Thesis 2011:09 on “Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD”.
8. Professor Sunilkumar Shinde, Mustansir Hatim Pancha / *International Journal of Engineering Research and Applications (IJERA)* ,”Comparative Thermal performance of shell and tube heat Exchanger with continuous helical baffle using “, Vol. 2, Issue4, July-August 2012.
9. KHAIRUN HASMADI OTHMAN, ” CFD simulation of heat transfer in shell and tube heat exchnager”, A thesis submitted in fulfillment for the award of the Degree of Bachelor in chemical Engineering (Gas Technology),April 2009.